

How to DIY design PCB layout by yourself

The printed circuit board is an essential component in electronic devices, serving as the foundation for various functions. It has been widely used and continues to be a vital element in electronic design. Apart from its cost-effectiveness and versatility, one significant advantage of PCB is the ability to customize their layout, making them accessible to users of all levels of expertise. In this TechSparks article, we will provide you with a **step-by-step tutorial on PCB design**, enabling even beginners to engage in DIY projects.

Table of Contents



1. Start with a schematic design
2. Build PCB Layout
3. Run Design Rule Check

Latest Post

Random Post

Popular Post

What is CEM-1 Material in PCB Board

August 30, 2023

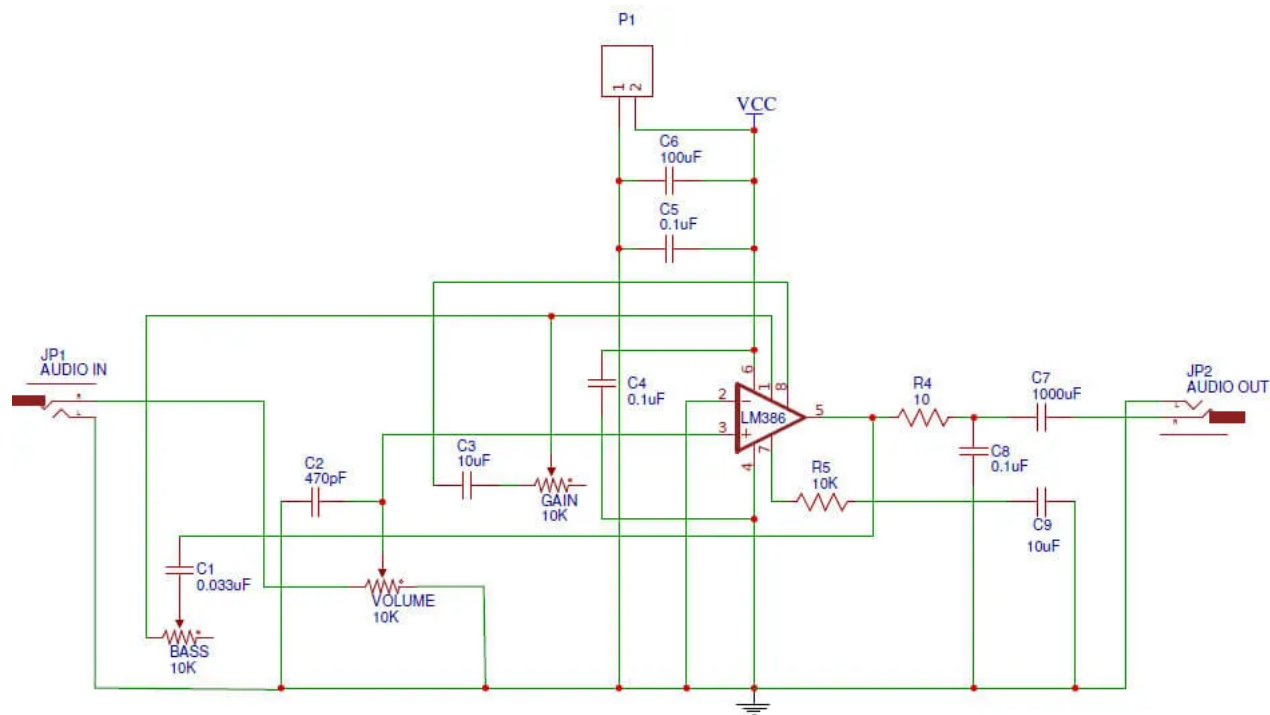
PCB Surface Finish Types and Selection Guidelines

August 30, 2023

Analysis and Resolution of the Global Chip Shortage Event

August 28, 2023

Start with a schematic design



For beginners, TechSparks recommends initiating the design process with the **PCB schematic**. The schematic serves as the foundation for circuit design, illustrating the interconnection relationships among [electronic components](#) and circuit functions. Guided by the schematic, the designer strategically plans and arranges the placement of electronic components on the PCB.

Here we take **EasyEDA** as a case to explain, this is a PCB design software with a wide range of applications, whether it is an enterprise, a student, or a hobbyist!

After logging in to the software, as shown in the figure, there is a button option of “New Project” in the upper right corner, click and enter

Comprehensive Knowledge of PCB
Copper Thickness

August 27, 2023

Immersion Silver PCB Surface Finish

August 26, 2023

Immersion Tin PCB Surface Finish

August 25, 2023

the upper right corner, click and enter.

After entering the project, as shown in the figure below, click the “New Schematic” button option.

Now you can see a blank artboard, which contains many function buttons. Knowing the purpose and shortcut keys of the function keys before starting to draw will help to quickly realize the PCB board design.

In schematic design, it is advisable to begin by placing all the required schematic symbols on the canvas before drawing the circuit. This approach allows for a more strategic and efficient connection process. The schematic symbols can be accessed from the “Library” located on the left side of the design software, as depicted in the accompanying image. In addition to the default symbols provided by the software, there are also user-generated “libraries” available, which greatly simplifies the design process for newcomers. Utilizing these libraries can help avoid unnecessary complications and streamline the design workflow.

When utilizing EasyEDA for PCB schematic design, it is essential to associate each schematic symbol with its corresponding package. The package, or footprint, determines the physical dimensions of the component and the placement of the copper pads or vias on the board. It is crucial to decide on the components to be used at this stage.

In EasyEDA, the schematic symbols available in the library are already pre-matched with their respective footprints. This means that when you select a symbol in the schematic, the corresponding [PCB footprint](#) is automatically assigned. However, if you require components with different sizes or styles, EasyEDA allows for easy customization. You have the flexibility to modify the physical dimensions and packaging style of the component to suit your specific design requirements.

To modify the footprint associated with a schematic symbol in EasyEDA, you have the option to search the User-Generated library for a suitable footprint that matches the component you are using. Once you find a package that meets your requirements, you can bookmark it by clicking the designated heart icon. This allows you to save the footprint for future use and easily access it in your subsequent circuit board design.

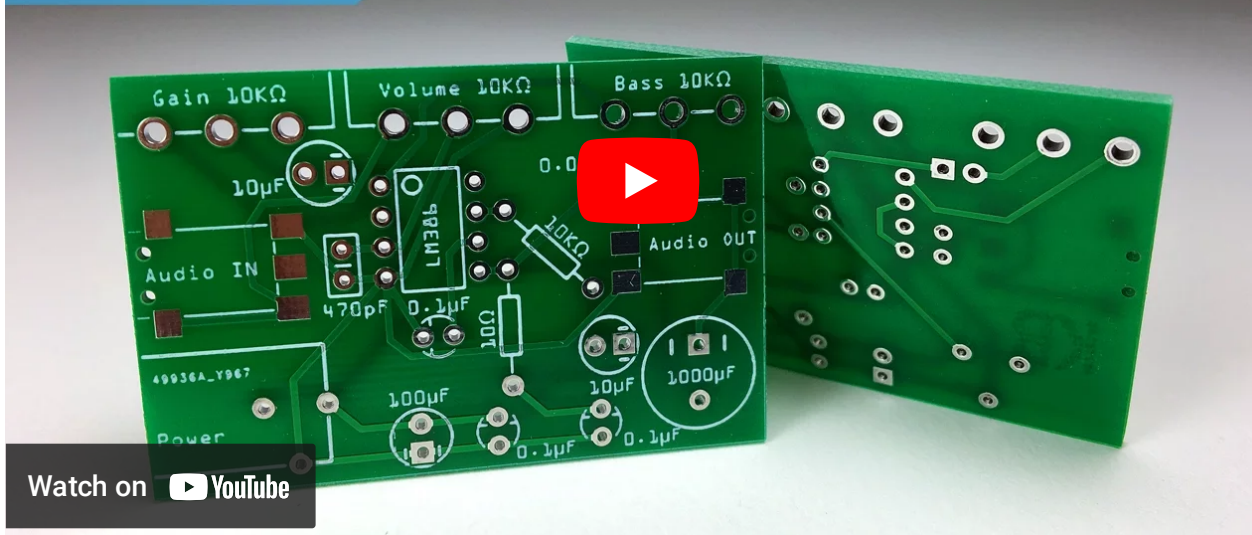
Copy and paste the desired component footprint into the schematic editor.

After completing the drawing of the circuit schematic diagram according to the process, import it into the editor. The following is a video explanation of the schematic diagram design:





TUTORIAL



Build PCB Layout

Before starting a PCB layout design, it is advisable for beginners to refer to the PCB design guide to

avoid common mistakes and ensure a smooth process. EasyEDA offers a convenient feature that allows you to convert a schematic diagram into a layout. Simply open the schematic diagram in EasyEDA and click on the “Convert Project to PCB” button located on the upper function bar. This enables a seamless transition from the schematic to the PCB layout stage.

Once you have finished the schematic design and transitioned to the layout editor, the footprint information linked to the schematic symbols will be automatically transferred and assigned to the corresponding components. This eliminates the need for manual re-entry or adjustment of package details, resulting in time savings and minimizing the risk of human errors. Upon completing the switch, you will notice that each component already possesses an associated footprint, and its physical dimensions, pad locations, and via placements align with the symbols in the schematic.

Within the layout editor, you will notice the presence of thin blue lines known as **“Ratsnest” lines**. These lines serve as a visual guide and reference point for component placement and wire routing during the layout process. They depict the connectivity between components and illustrate their physical arrangement on the board. The Ratsnest lines indicate the specific areas where you need to route traces on the circuit board layout, aligning with the wiring connections established in the schematic design.

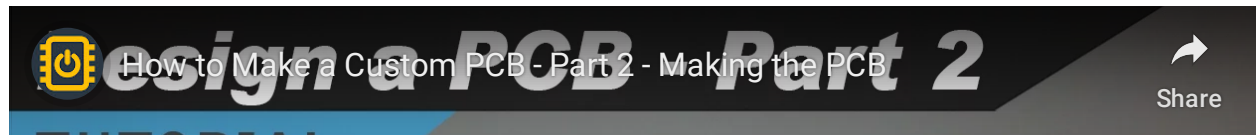
Similar to the schematic design process, the layout process involves arranging components and then establishing the necessary wiring connections. To ensure a successful layout, it is essential to conduct thorough research and consider the specific requirements of the application. For instance, certain components may need to be placed in specific positions to optimize their functionality. For example, in amplifier circuit design, placing power supply decoupling capacitors near the chip can help reduce electromagnetic noise.

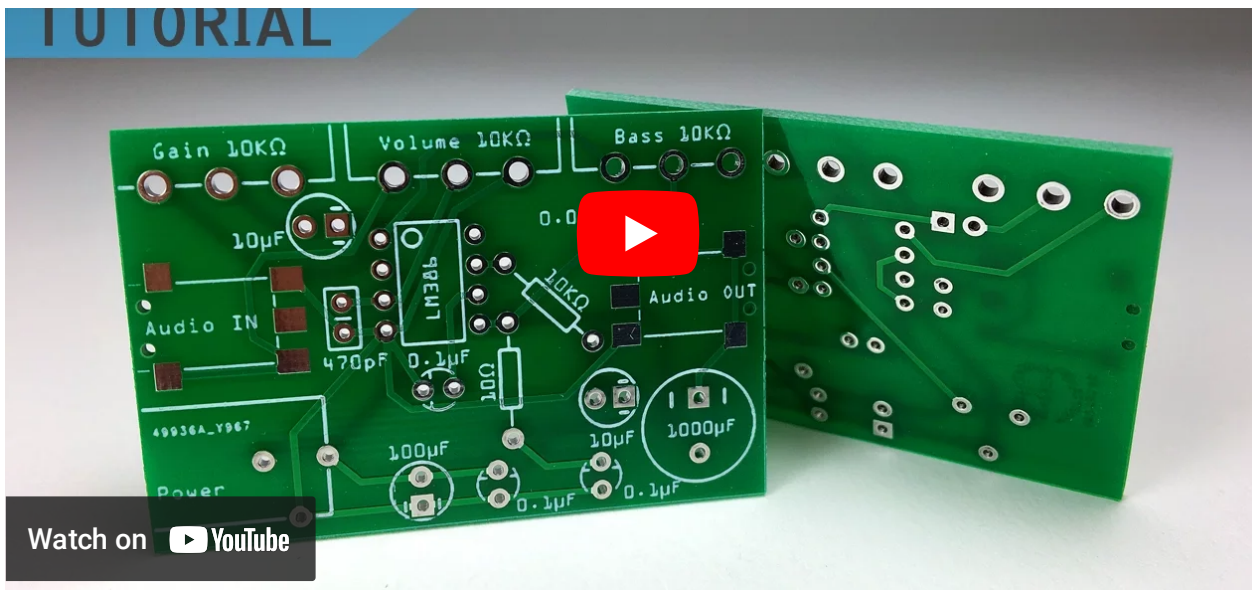
Once the components are placed, you can use the initial “Ratsnest” lines as a guide for routing the traces. However, it is important to note that these lines may not always indicate the optimal routing paths. Therefore, it is recommended to review the schematic to verify the correct connections.

Additionally, some layout software offers both automatic and manual routing functions, allowing you to choose based on the circuit’s complexity during the DIY PCB design process. Generally,

manual routing is recommended for complex circuits, while automatic routing can be utilized for simpler circuits to expedite the design process. After the initial routing is completed, it is advisable to revisit the PCB design layout and make fine adjustments as necessary.

Here is a detailed video of drawing traces in the PCB design process flow:





Run Design Rule Check

So far the design has been basically completed. Before sending the design files to the PCB fabrication supplier, it is crucial to perform design rule checks as a necessary step in the PCB design procedure. These checks ensure the design's integrity by identifying any potential faults such as overlapping components or inadequate trace distances. To initiate the design rule checking, click the "Design Manager" button located in the function window on the right-hand side. Within the Design Manager, you will find the option to perform the design rule check.

Within the DRC function, any design issues that arise will be presented as errors in the designated “DRC Errors” folder. By selecting one of the listed errors, the specific trace or component associated with the problem will be highlighted within the PCB view. This invaluable feature enables designers to pinpoint and address the precise location of any design irregularities.

To customize the design rule checking settings, you can access the designated area by selecting the drop-down menu located in the upper right corner. From there, navigate to the “Miscellaneous” section and click on “Design Rule Settings.” This will allow you to define and specify your own set of rules for the design rule checking process.

Upon opening the designated window, you will be presented with a comprehensive set of options to define various design rules. These rules encompass parameters such as trace width, spacing between traces, and other relevant criteria. This customizable interface empowers you to establish specific guidelines and constraints for your PCB design.

Once you have thoroughly verified the usability and functionality of your PCB design and addressed any design issues, your work is ready to be finalized. At this stage, you can proceed with sending the design files to an online PCB manufacturer for fabrication. By doing so, you can entrust the manufacturing process to experienced professionals who will transform your design into a physical PCB. This marks the completion of your PCB design journey, allowing you to move forward with confidence in the realization of your electronic project.

Resource

Useful Links

Social Link

[PCB Terminology](#)

[Sitemap](#)

